Numerical Analysis of Heat Transfer and Thermal Performance Analysis of Surface with Circular Profile Fins

Sunil Chamoli, Ranchan Chauhan, N.S. Thakur
Centre for Excellence in Energy and Environment
National Institute of Technology, Hamirpur H.P. India

***Abstract*-In the present work a Computational Fluid Dynamics (CFD) study was conducted to investigate the heat transfer and friction loss characteristics in a horizontal rectangular channel having attachments of circular profile fins over one of its heated surface. The Reynolds number based on the flow averaged inlet velocity and the hydraulic diameter, ranged from 5000 to 30,000. The circular profile fins considered are of 5 cm height and diameter of 2 cm dimensions mounted on a heating surface vertically. Reynolds number, fin arrangement and fin pitch in the flow direction are the numerical parameters. The in-line and staggered fin arrangement was studied for one-fixed span wise (Sx/d = 3) and four different stream wise (Sy/d = 3, 3.75, 5 and 7.5) distances. Different turbulent models have been used for the analysis and their results are compared. Realizable k-ε model based results have been found a good agreement and thus this model is used to predict the heat transfer and friction factor in the duct. The overall enhancement ratio has been calculated in order to discuss the overall effect of fin spacing and operating parameters. A maximum value of enhancement ratio is found 1.2 for the range of parameters investigated.**

Keywords-Finned surface; Thermal performance; CFD; Heat transfer; Friction; turbulence

NOMENCLATURE

S P itch in m

d Diameter of the fin in m

N_{tot} Total number of fins

Q Heat transfer rate

m Mass flow rate of air in kg/s

C_p Specific heat of air in J/ kg-o

T temperature of fluid in ^oC

h Heat transfer coefficient in W/m2

A Area in m2

T_s Average plate temperature.

W Width of the plate in m

L Length of the finned plate in m

H Height of the fin in m

P Pressure in Pascal

D_h Hydraulic diameter in m

V Mean inlet velocity in m/s

ρ Fluid density in kg/m3

Subscript

*Co*n Conduction

Conv Convection

Rad Rad iat io n

in, out Inlet and Ou tlet

x, y Span-wise and Stream-w is e

Δ Drop

s Surface

av Average

Ⅰ. INTRODUCTION

Straight fin with concave parabolic profile provides maximum heat dissipation for a given profile area [1]. As the concave shape is costly and difficult to manufacture, the rectangular profile is preferred even though it is not utilize the material most efficiently [2]. For example, in a study performed by Tahat et al [3], pin fins were employed on the heating surface in a rectangular channel; and Bilen et al. [4] investigated the heat transfer and friction loss characteristics of a surface with cylindrical fins arranged both inline and staggered in a channel having rectangular section. The maximum amount of heat transfer occurred at S_y/D=2.94. A thermal performance analysis is also worthwhile for the evaluation of net energy gain. One of the ways to evaluate the heat transfer performance is the comparison of the heat transfer coefficients at the constant pumping power [4-6]. Many studies have been done for different types of fin arrays, for example one reported in [7], but still there is lack of knowledge of the forced convection heat transfer from a surface with vertical mounted hollow rectangular profile fins. The experimental analysis was carried out to study the heat transfer and friction loss characteristics; the relation was deprived for nusselt number and friction factor [8]. CFD analysis has been done by various investigators to enhance heat transfer from a surface. CFD analysis of solar air heater provided with different metal ribs of circular, square and triangular cross-section with 60^0 inclination to air flow was studied [10] and found 30% increase in heat transfer. Ozceyhan et al. [11] conducted numerical

investigation on heat transfer enhancement in tube with the circular cross-sectional rings. Iaccarino et al. [12] studied effect of thermal boundary Conditions in numerical heat transfer predictions in rib-roughened passages. CFD analysis of heat Transfer and flow analysis due to roughness in the form of ribs was carried out by these investigators using 2-D models only.

In the present work the study of fluid flow and heat transfer in a rectangular channel with circular vertically mounted fins using computational fluid dynamics has been carried out which reduces time and cost . The fin number and the distance between the fins are given in the Table 1, and the arrangement of the fins in the test section is shown in Fig. 1 and 2.

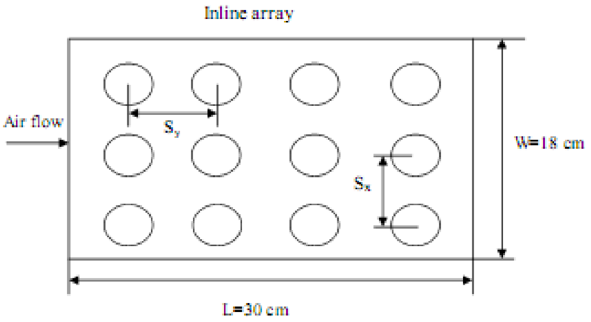

Fig 1: Inline arrangement for $S_y/d=3.75$

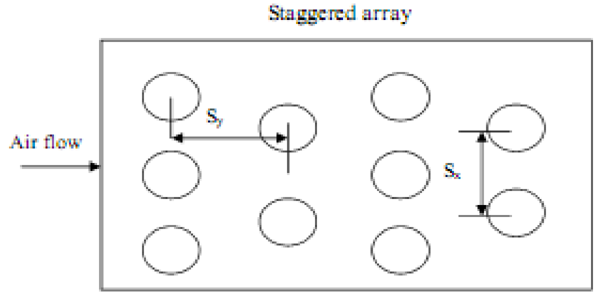

Fig 2: Staggered arrangement for Sy/d=3.75

Table 1: Number of circular profile fins for 2cm diameter and $S_x/d=3$

S_y/d	3	3.75	5	7.5
N_y	5	4	3	2
N_x	3	3	3	3
N_{tot}(in-line)	15	12	9	6
N_{tot} (Stag)	13	10	8	5

II. ANALYSIS

A. Methodology

The heat transfer mode of interest of this system is conduction, convection, radiation through the air. The magnitude of each mode depends on the temperature of pin fin array base, the geometry and the flow rate. Heat balance for the whole system is

$$\dot{Q}_{Tot} = \dot{Q}_{Con} + \dot{Q}_{Conv} + \dot{Q}_{Rad} \quad (1)$$

$$\dot{Q}_{Conv} = \dot{m} c_p (T_{Out} - T_{in}) \quad (2)$$

$$\dot{Q}_{Conv} = h_{av} A_s \left[T_s - \left(\frac{T_{out} + T_{in}}{2} \right) \right] \quad (3)$$

Surface area As= W x L+ Π d Nx Ny (H - d/4)

The total radiative heat losses from a similar test surface would be about 0.5% of the total electrical input. This was also reported by Tahat et al [3].

Since in the experimental setup the sidewalls are adiabatic walls the conductive heat loss from the side walls can be neglected in comparison to that from the bottom surface of the test section. Moreover, as mentioned above, the bottom surface of the test plate which is not exposed to the flow, have been insulated, thus it can be thought that the conductive heat loss from the heated plate will be mainly from its back surface into the environment through the bottom wall of the channel.

B. Solution domain

Fins arrangements both in line and staggered at different stream-wise pitches (Sy= 3, 3.75, 5, 7.5) and a fixed span wise pitch (Sx=3) has been considered. The duct used for CFD analysis having the height (H) of 10 cm and width of (W) of 18 cm. A 2.8 cm wooden Planck is considered at the sides of the duct for insulation. A uniform heat flux of 1000W/m2 was considered for analysis. The fins are considered on the heated plate and other sides are considered as smooth surfaces.

The pressure drops over the test section in the model were measured. The pressure drop were arranged in non dimensional form by using the following equation

$$f = \frac{2\Delta p}{\frac{L}{D_h} \rho V^2} \quad (4)$$

C. Grid

The chosen geometry is such that secondary flows condition are bound to occur thus using a 2D domain is ruled out .Thus 3D solution domain and grid were selected. In order to examine flow, boundary layer phenomena and heat transfer at the fins surface finer meshing is done at these locations. In the other region coarser mesh is used. For the relevance of the solution, all simulations were repeated with twice the number of the grid points in each spatial direction. No noticeable differences in the solutions were observed.

D. CFD analysis

Under the present study, commercial code FLUENT version 12.0.16 was used. As the secondary flows takes

place thus 3D model has been setup instead of 2D model to simulate flow and heat transfer.

The assumptions were made in mathematical model:

1. The flow is considered fully developed, turbulent and three dimensional.
2. The thermal conductivity does not vary with temperature.
3. The duct wall and fins material is homogeneous and isotropic.

The working fluid, air is assumed to be incompressible for the operating range of finned duct since variation is very less. The mean inlet velocity of the flow was calculated using Reynolds number. Velocity boundary condition has been considered as inlet boundary condition and outflow at outlet. Second order upwind and SIMPLE algorithm were used to discreitize the governing equations.

The FLUENT software solves the following mathematical equations which governs fluid flow, heat transfer and related phenomena for a given physical problem. The equations are the

Continuity equation

$$\frac{\partial}{\partial t}(\iiint_{v} \rho dv) + \iint \rho V ds = 0 \tag{5}$$

Momentum equation

$$Fx = (-\frac{\partial p}{\partial x} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z})dxdydz + \rho f_{x} dxdydz \tag{6}$$

E. Fins geometry and range of parameter

Aluminum fins of diameter 2cm and height of 5cm were used and number of fins on the heater plate depends on the pitch. The fixed pitch across span direction (Sx/d=3) and variable pitch (Sy/d= 3, 3.75, 5, 7.5) along steam wise direction is considered. The number of fins for inline, staggered fin arrangement pattern at different pitches is shown in table 1. The range of Reynolds number is taken from 5000 to 30000.

III. RESULTS AND DISCUSSIONS

A. Selection and validation of the model

Different models namely standard k-epsilon model, realizable and RNG k-epsilon model have been tested for smooth duct having same dimension of the computational domain for validity of the models The results obtain by different models have been compared with result correlated with nusselt number for smooth channel.

$$Nu_{s} = 0.036 \, \mathrm{Re}^{0.8} (\frac{D_{h}}{L})^{0.055} \tag{7}$$

Fig 3 shows the variation of nusselt number with Reynolds number for different turbulence models and the results compared with the correlation available in the literature for smooth duct. It has been observed that the results obtained with realizable k-epsilon model are in good agreement with the results from the correlation.

B. Temperature Contours, Velocity contours and vectors

The flow and heat transfer characteristics get affected in the flow direction due to fins provided. Fig 4 shows numerical solution of the flow at Sy/d=3.75 and Sx/d=3. It can be seen from the figure that the flow for the lower Sy/d has to negotiate a serpentine path through the pin fin arrays whereas for the higher Sy/d, the flow is able to pass through the array without much resistance. It was observed that the fluid flow at larger Sy/d has minimal interaction with the pins surfaces. The phenomenon contributes greatly to the reduction of pressure drop.

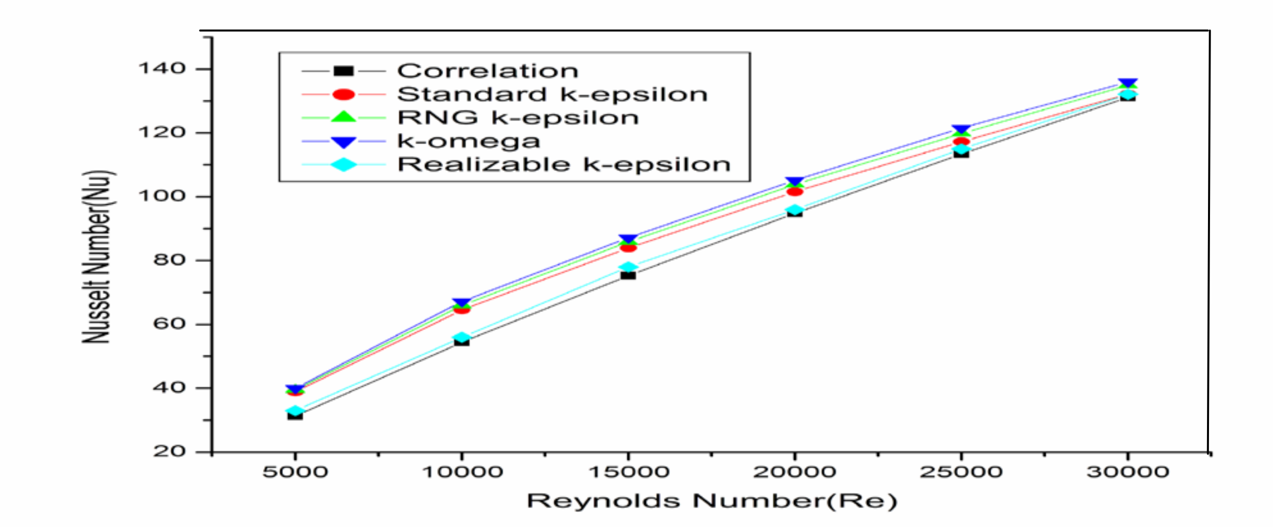

Fig: 3: Turbulence model comparison for smooth duct

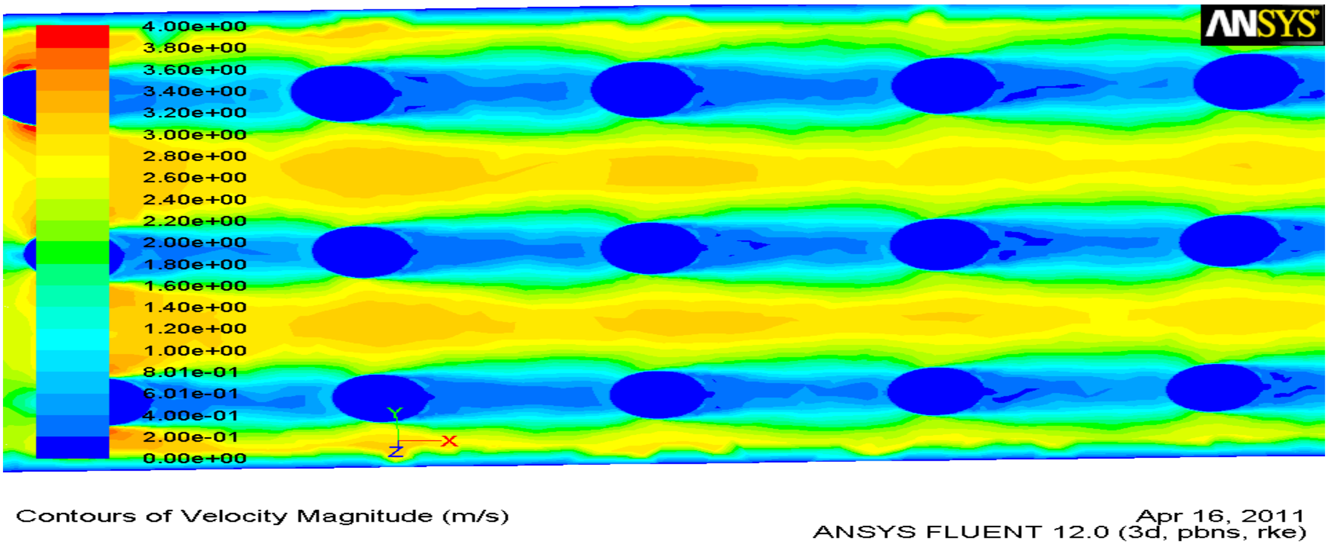

Fig 4: Velocity contour at Sy/d=3.75

From the Fig. 6 it is found that more vortices are making in the flow direction thus creating turbulence and the nusselt number increases. The flow is distributed by blockage due to presence of fins the first fins along the flow direction put much obstruction for fluid flow in inline arrangement but for staggered flow is much smooth. The shedding of vortices also causes additional loss of energy resulting in increased friction factor. Thus nusselt number and friction factor for finned duct is higher compared to smooth duct. Fig 5 and Fig 6 shows static temperature on the finned surfaces. It shows that temperature of air increasing along length of the duct, as the primary heated air moves along the heated fins. Hence the temperature at the base of the fin is higher. As the heated primary air is swapped by the fin then lower temperature secondary air comes in contact with hot surface result in to increasing heat transfer coefficient

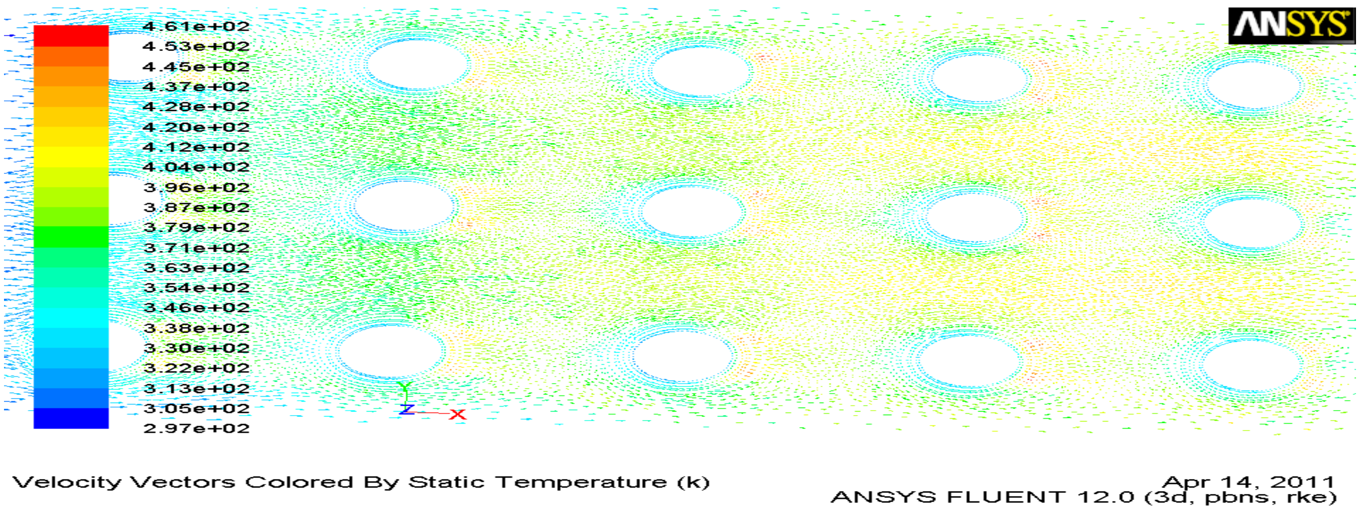

Fig 5: Velocity vectors colored by static temperature at Sy/d=3.75

C. *Thermo-hydraulic performance of finned duct:*

Fig 8 and Fig 9 shows the pin fin array Nusselt number as a function of Reynolds number for inline and staggered arrangement at all Sy/d considered in the study. It is seen that the pin spacing along the stream wise direction have influence on the rate of average heat transfer. Increasing or decreasing the pin distance would reduce or increase the number of pin rows of a fixed base length. Thus by increasing or decreasing the Sy/d with the pin diameter and length being kept constant, the total surface area in contact with the fluid will decrease or increase accordingly. It can be seen from Fig that the average Nusselt number is the lowest when the Sy/d =7.5 and the highest average Nusselt number is at Sy/d = 3. While there is an appreciable increase in average Nusselt number when the Sx/d is reduced from Sx/d = 5 to Sx/d = 3 and from Sx/d =7.5 to Sx/d =5. The results of the staggered arrangement are found good for all pin spacing compared to inline arrangement and enhancement originated from the increasing intensity of the turbulence and better mixing of fluid.

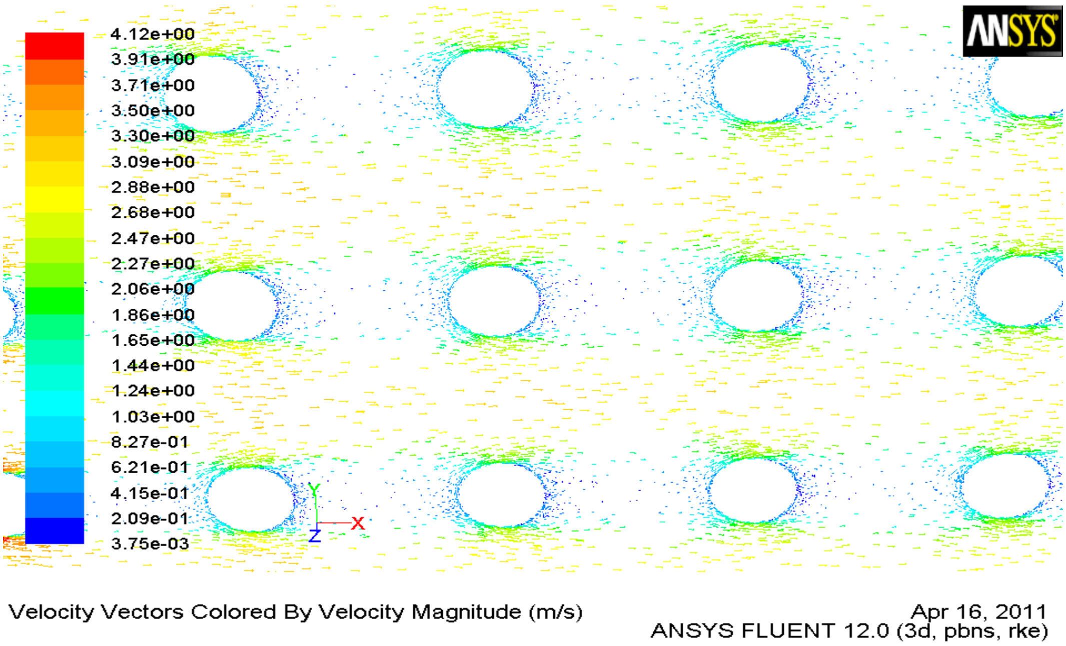

Fig 6: Velocity vectors colored by velocity magnitude at Sy/d=3.75

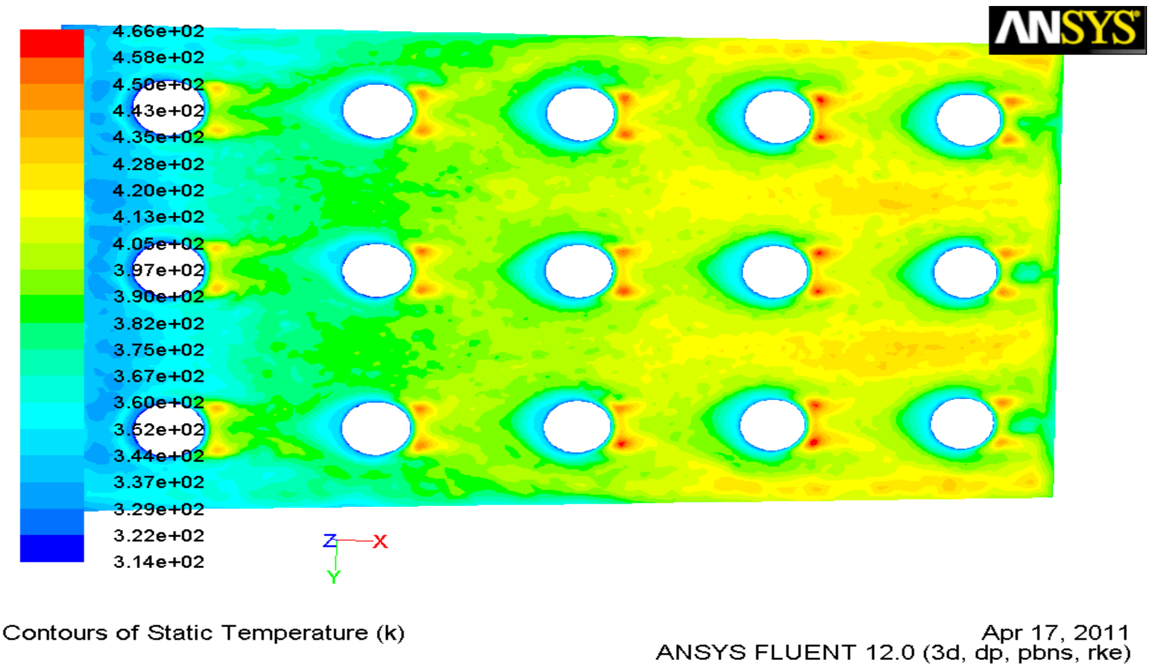

Fig 7: Contours of temperature at Sy/d=3.75, Re=5000

Fig. 10 and Fig. 11 shows a plot of friction factor as a function of Reynolds number for inline and staggered arrangement at all the Sy/d being studied. The plots clearly show that the friction factor varies significantly with changes in Sy/d. The entire configuration has similar characteristic and trends showing that the friction factor decreases with increasing Sy/d. When Sy/d is decreases, the smaller stream wise spacing shrink the entrance area to the fluid flow and thus fluid flow at higher velocity and thus increases the pressure drop to occur. On contrary, increasing Sy/d will open up the flow passage and flow is allowed to pass more smoothly through the array section with relatively less resistance. This explains the lower friction factor that was observed for higher Sy/d. The values of pressure drop for inline arrangement at lower Reynolds number is higher than the staggered arrangement due to fact originated from more blockage to the fluid flow in inline arrangement.

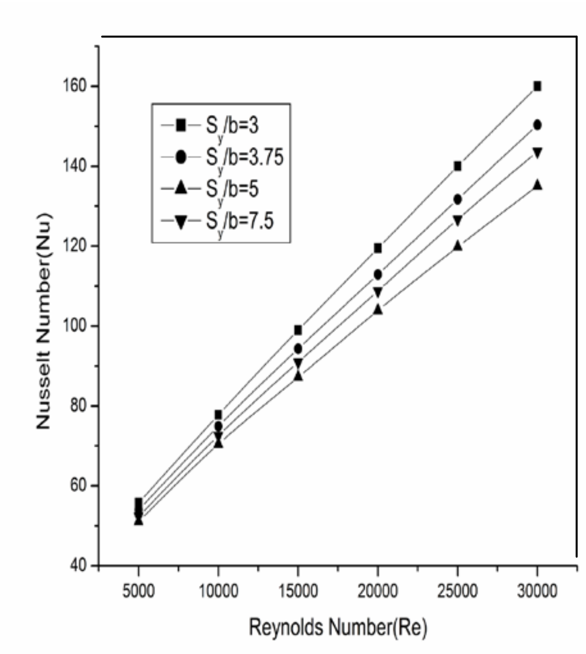

Fig.8: Variation of Nusselt number with Re for inline arrangement

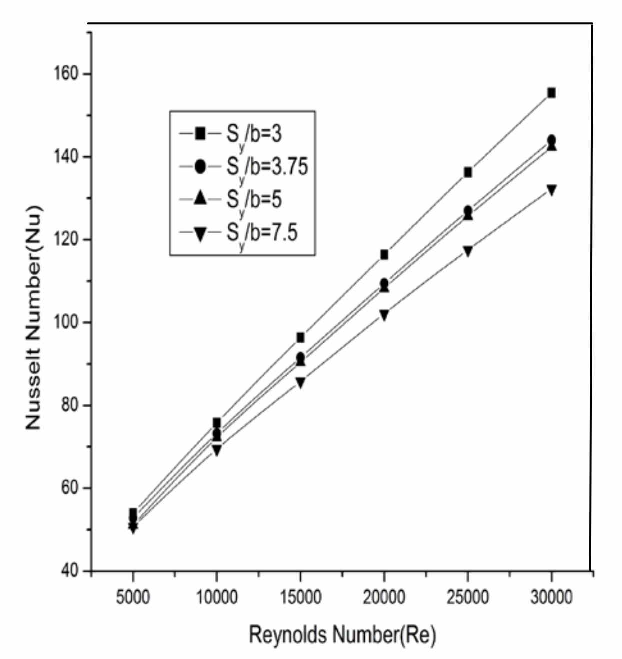

Fig.9 Variation of Nusselt number with Re for staggered arrangement

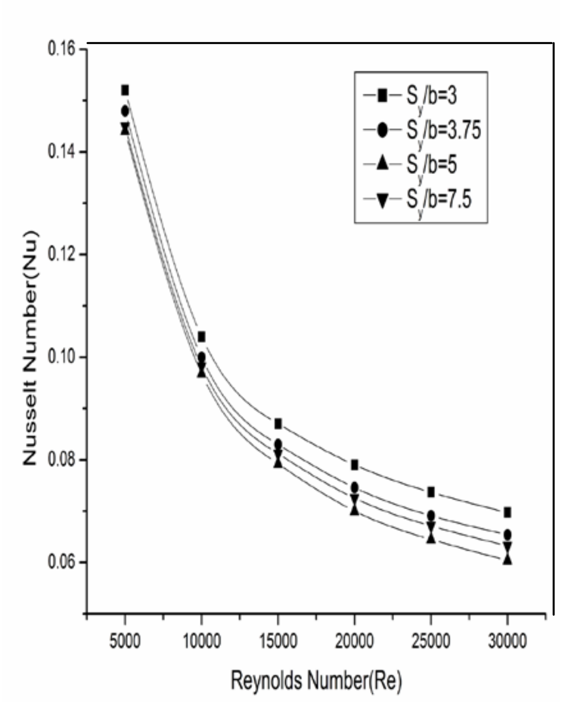

Fig.10 Shows variation of Friction factor

with Reynolds number for inline arrangement

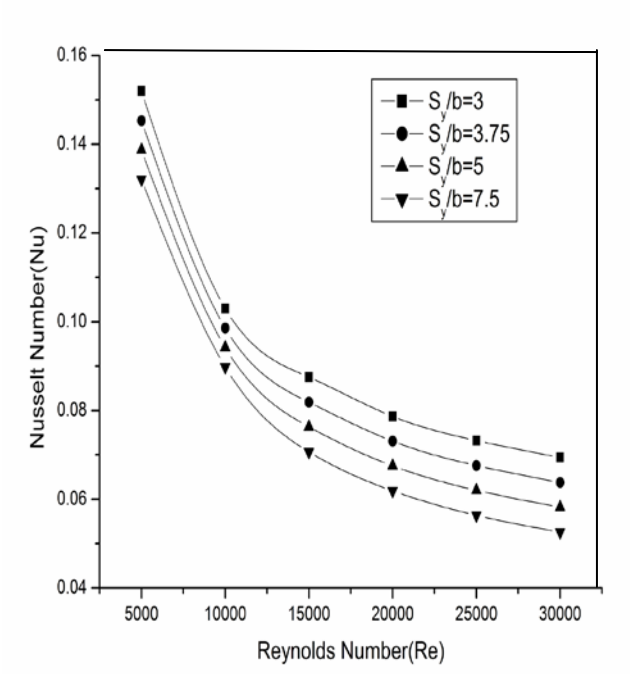

Fig.11 Shows variation of Friction factor with

Reynolds number for staggered arrangement

Overall enhancement ratio is defined by the following expression [13].

$$\text{Overall enhancement ratio}=\frac{\dfrac{Nu_r}{Nu_s}}{\left(\dfrac{f_r}{f_s}\right)^{1/3}} \qquad (8)$$

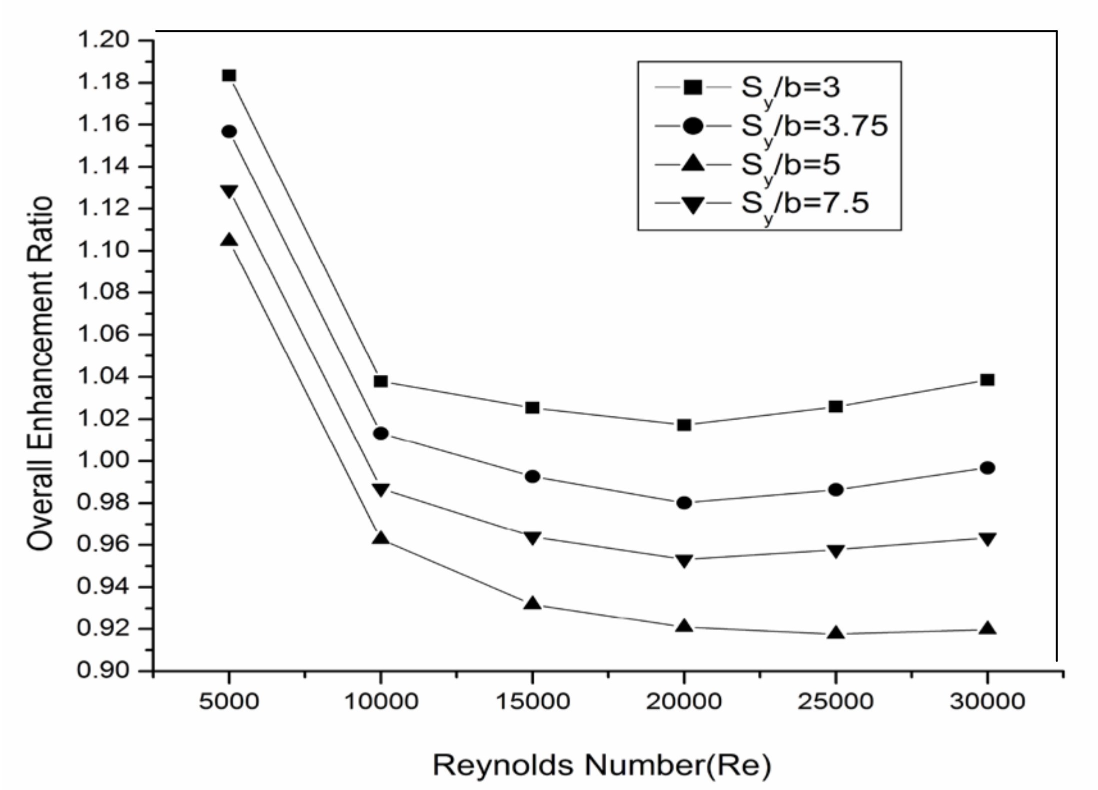

Fig. 12: Shows variation of overall enhancement ratio with Reynolds number

Fig. 12 shows overall enhancement ratio of finned duct for reynolds number range from 5000 to 30000. It has been found that overall enhancement ratio at stream wise pitch Sy/d=3, 3.75 is greater than unity for operating range of reynolds number. The overall enhancement ratio decreases sharply up-to 10000 reynolds number and a little decrement was observed from 10000 to 30000 reynolds number. Further overall enhancement ratio found maximum for stream wise pitch of Sy/d=3 at 5000 reynolds number

IV. CONCLUSION

An attempt has been made to carry out CFD based analysis to fluid flow and heat transfer characteristics of a hollow rectangular duct attached with circular profile fins. Combined effect of swirling motion, detachment and reattachment of fluid which was considered to be responsible in the increase in the heat transfer rate has been observed during CFD analysis. Nusselt number found increase with increase in Reynolds number while Friction factor decreases with increase in Reynolds number at all set of stream wise pitches for both the inline and staggered arrangement of fins geometries.

CFD results have also been validated for smooth duct and different CFD turbulence model has been compared with empirical correlation for smooth duct. Among the entire models realizable k-epsilon model found good agreement with empirical correlation.

Overall enhancement ratio with a maximum value of 1.2 is found for the stream-wise pitch of Sy/d=3 at 5000 Reynolds number.

REFERENCES

[1] Q.D. Kern, D.A. Kraus, Extended Surface Heat Transfer, Mc Graw-Hill, New York, 1972.

[2] L. T. Yu, C.K. Chen, Application of Taylor transformation to Optimize rectangular fins with variable thermal parameters, Applied Mathematical Modeling 22 (1998) 11–21.

[3] M. A. Tahat, R. F. Babus Haq, S.D. Probert, Forced steady state Convections from pin fin arrays, Applied Energy 48 (1994) 335– 351.

[4] K. Bilen, U.Akyol, S.Yapici, Heat transfer and friction Correlation and thermal performances analysis for a finned surface, Energy Conversion and Management 42 (2001) 1071–1083.

[5] P. Dutta, S. Dutta, Effect of baffle size, perforation, and Orientation on internal heat transfer enhancement, International Journal of Heat and Mass Transfer 41 (1998) 3005–3013.

[6] J. J. Hwang, T. M. Liou, Heat transfer and friction in a low-aspect-Ratio rectangular channel with staggered perforated ribs on two opposite flows, ASME Journal of Heat Transfer 117 (11) (1995) 843–850.

[7] K. Bilen, U. Akyol, S.Yapici, Thermal performance analysis of a tube finned surface, International Journal of Energy Research 26 (2002)321–333.

[8] A Ugur, K Bilen Heat transfer and thermal performance analysis of a surface with hollow rectangular fins, Applied thermal Engineering 26 (2006) 209-216

[9] S.V Karmare, A.N Tikekar, Analysis of fluid flow and heat transfer in a rib grit roughened surface solar air heater using CFD, Solar Energy, 84 (2010) 409-417

[10] Ozceyhan Veysel, Gunes Sibel, Buyu kalaca Orhan, Altuntop Necdet. Heat Transfer enhancement in tube using circular cross-sectional rings separated from wall. Applied Energy 2008; 85: 988–1001.

[11] Iaccarino G, O oiA, Durbin PA, Behnia M. Conjugate heat transfer predictions in two dimensional ribbed passages. International Journal of Heat and Fluid Flow 2002; 23: 340–5.

[12] Wang L, Sunden B, Performance comparison of some tube inserts. International Communication in Heat and Mass Transfer 2002; 29(1); 45-56